EXPERIMENTAL AND NUMERICAL STUDY OF THE EFFECT OF THE DOWNSTREAM SPILLWAY FACE’S ANGLE ON THE STILLING BASIN’S ENERGY DISSIPATION

Thulfikar Razzak Al-Husseini
Associate Professor, Civil Eng. Dept., University of Al-Qadisiyah, Al-Qadisiyah, Iraq

Hayder A. Al-Yousify
Assistant Lecturer, Civil Eng. Dept., University of Al-Qadisiyah, Al-Qadisiyah, Iraq

Munaf A. Al-Ramahee
Lecturer, Civil Eng. Dept., University of Al-Qadisiyah, Al-Qadisiyah, Iraq

ABSTRACT
A new experimental and numerical study was presented to investigate the effect of downstream spillway face’s angle on the stilling basin’s energy dissipation (ED) efficiency. Six composed physical plywood models with downstream slopes of spillway face ranged from 30° to 55° for the spillway was built and used in the experimental tests. The hydraulic parameters of the flow were measured and the ED has calculated accordingly. The experimental results showed that the relative energy dissipation (RED) of the flow on the stilling basin decreased when the discharge was increased. On the other hand, the energy dissipation increased when the angle of the downstream face of the spillway was decreased. These models were simulated using 3-D Computational Fluid Dynamics (CFD) tool through (Flow-3D V.10.0.3) software. The comparison between the experimental work and numerical model indicated a significant convergence between them with R2 equals to 0.99.

Key words: Spillways, ED, Stilling basin, CFD, Flow-3D.


1. INTRODUCTION
Energy dissipation (ED) efficiency considered one of the most problems that were studied on various structures such as stilling basins, hydraulic drops, etc. Many studies have been
conducted to study this problem. The main goal of these studies was to increase the efficiency of these hydraulic structures to dissipate more energy and therefore to prevent the failure.

Amorim et al. presented a numerical and experimental hydraulic jump study using a physical model for the prototype Colombia power plant basin and CFD model [1]. They concluded that the CFD is an effective tool for representing the flow over stilling basin [1]. Aboul Atta et al. also presented an experimental study to improve the roughness of stilling basin using a T-shape instead of traditional one [2]. The suggested T-shape used in their work decreased the length of the hydraulic jump [2]. Taebi et al. designed a riprap for stilling basin to increase the tail water and control the scouring [3]. Smith and Klassen suggested a hydraulic design of two-stage stilling basin which was used for high ED of structures. The suggested two-stage stilling basin was proved to be effective for the two hydraulic jumps occurred over stilling basin [4].

On the other hand, Hinge et al. presented a new design for stepped weir which is located at the end of stilling basin to improve its efficiency. The modified design prevented the hydraulic jump that sweeps out the stilling basin [5]. Shahmirzadi et al. presented an assessment for the ED of in-ground stilling basin and a correlation between flow patterns and the in-ground stilling basin. They concluded that there is a significant correlation between the flow pattern and the geometry of in-ground stilling basin [6]. Tiwari et al. conducted an experimental study for the new design of USBR VI stilling basin model for pipe outlet. The new design was more efficient and more economic than other models by changing the location and geometry of impact walls and splitter blocks [7].

Kantoush and Sumi presented a study on the influence of stilling basin geometry on sediment transport and flow pattern at flood mitigation dams. The study developed a new method of the optimum design of stilling basin that dissipated more energy and deposited minimum sediment [8]. Bestawy suggested new shapes of baffle piers that can be used in stilling basin to dissipate more energy. In this study, 14 shapes of baffle piers were introduced as new models of stilling basin. Sand was used downstream of stilling basin and the dimensions of scour were measured to specify the ED [9].

Tiwari and Goel conducted an experimental study to investigate the effect of end sill on the performance of the stilling basin. New physical models were suggested and tested for three Froude No. which are Fr = 1.85, 2.85, and 3.85. The results have shown that the triangular shape of end sill for the same Froude No. gives the best performance compared with other shapes [10]. Verma et al. presented an experimental study for a new design for stilling basin. Several devices such as impact wall, grid, weir wall, stepped wall, wedge-shaped blocks, and sloping end sill were used to study their effect on stilling basin. Each model was tested at Froude No. = 4.89 and the scour downstream the stilling basin was measured. The study showed that the wedge shapes blocks were the best of dissipating the energy compared with the other devices [11].

Over the past 30 years, numerical modeling techniques have been rapidly developed as computational power and enhanced to the point where numerical solutions are now possible for many applications. This development led to a widespread use of numerical modeling as a standard design tool in many engineering disciplines. Zhenwei and Zhiyan studied the simulated of flow characteristics in a whole spillway based on the VOF method by FLUENT software [12]. The numerical computation results of the surface elevation, pressure and flow velocity along the spillway in two schemes fit the experimental results well, and the difference of the average velocity between calculated and experimental results was less 6%.
Computational fluid dynamics (CFD) is a type of numerical modeling that is used to solve problems involving fluid flow. Since CFD can provide a faster and more economical solution than physical modeling, hydraulic engineers are interested in verifying the capability of CFD software. Although some literature showed successful comparisons between CFD and physical modeling, a more comprehensive study would provide the required confidence to use numerical modeling for design purposes [13]. In this paper, a new experimental and numerical study was presented to investigate the effect of spillway angles on the stilling basin using six angles through six physical models.

2. EXPERIMENTAL APPROACH
The experimental tests were performed at the fluid laboratory in the College of Engineering of Al-Qadisiyah University using a rectangular ARMFIELD flume. The dimensions of the flume are (2.45×7.5×0.25) m (L,W,H). A sharp-crested weir with 2.5 cm high was constructed at the end of the flume to create a hydraulic jump. Figure 1 represents the typical dimensions of the models that used in this research.

The flow upstream depth was measured at a location more than (9 \( y_c \)) upstream of the spillway, where \( y_c \) is the depth of flow over the crest of the spillway. Discharges were measured using a calibrated flow meter located at the channel outlet. The maximum discharge of the flume was 2.64 ℓ/sec. The water surface levels were measured at different locations using an accurate point gauge reading up to 0.1 mm.

Relationships of the energy, RED, the length of the crests, and radius of curvature of upstream face in all models were used to design the six models as illustrated below [15] [16].

\[
E_1 = y_1 + \frac{aV_1^2}{2g} \tag{1}
\]

\[
E_2 = y_2 + \frac{aV_2^2}{2g} \tag{2}
\]

\[
\frac{\Delta E}{E_1} \% = \frac{E_1 - E_2}{E_1} \% \tag{3}
\]

\[
\frac{L_{cr}}{H_t - h_s} > (1.5 - 3.0) \tag{4}
\]

\[
R = 0.2 (H_t - h_s) \tag{5}
\]

\( E_1, E_2 \) are upstream and downstream energy (m), respectively; \( V_1, V_2 \) are the upstream and downstream velocity of spillway (m/sec), respectively; \( y_1, y_2 \) are upstream and downstream water depth; \( \frac{\Delta E}{E_1} \% = \) RED between spillway upstream and stilling basin downstream; \( a \) is the kinetic correction coefficient, which equal to 1.1 for turbulent flow [13]; \( g \) is the gravity acceleration (m/s²), \( L_{cr} \) is the length of crest of spillway; \( H_t \) is the upstream total head above the channel bed (from maximum discharge); \( h_s \) is the spillway height above the channel bed; and \( R \) is the radius of curvature of upstream face.

Saint Anthony Falls (SAF) type of stilling basin was used in this study due to its wide discharges ranges as it was mentioned by previous study with the major design rules for this type [15].

Using equations (4) and (5) and the design rules [15], six compound models of spillway and stilling basin were designed and constructed from plywood. In this research, the height of
all models was selected to be 60% of the flume height, i.e. 15 cm to avoid overflow. All compound models were located at distance of 0.75 m from the entrance of flume to eliminate the turbulence. The six angles used in the models were (θ = 30°, 35°, 40°, 45°, 50° and 55°) for the downstream sloping face of the spillway. These angles were selected to be greater than the critical value θ = 27° that is defined by Chanson [16].

For each model, twelve runs were used and the hydraulic parameters were measured for each run that includes: the discharge (Q), upstream flow depth (y1) at the beginning of spillway, and the downstream flow depth (y2) at the toe of stilling basin. After the hydraulic parameters were measured, the ED of flow was calculated.

Figure 1 Dimensions of the used model

The maximum ratio of (y1/y_c) (where y_c is the depth of flow over the crest of spillway) corresponding to the compound models of the experimental study is equal to 27.17. This value is less than the maximum limit of (y1/y_c < 35) recommended by Chafi [17,18]. Figure 2 shows the experimental tests for the six compound models that were designed and used in this research.

Model No.1; θ = 30°  Model No.2; θ = 35°  Model No.3; θ = 40°  Model No.4; θ = 45°
Experimental and Numerical Study of the Effect of the Downstream Spillway Face’s Angle on the Stilling Basin’s Energy Dissipation

Figure 2 The six compound models used in the experimental program

3. ENERGY DISSIPATION ANALYSIS
From the experimental results shown in Figure 3, the RED was increased when the angle of the downstream face of the spillway was decreased and vice-versa. This is obvious in Model No.1 (θ= 30°) which has the least angle of downstream slope of the spillway and the greatest RED as shown in Figure 4.

![Figure 3 RED- Discharge relationship for the six compound models](http://www.iaeme.com/IJCIET/index.asp)
4. NUMERICAL METHODOLOGIES
4.1. Numerical Model Implementation
There are many programs that can be used for modeling CFD, such as FLOW-3D, ANSYS, and CFX. However, the FLOW-3D commercial software program was adopted in this research because it has sufficient tools and easy to implement.

The general model set-up for all spillway and stilling basin simulations that were conducted was quite similar. In each case, the viscosity and turbulence option was activated with Newtonian viscosity being applied to the flow along with the selection of an appropriate turbulence model. The results showed that there were only minimal differences in the data of interest in this study with different turbulence models applied, as long as the more advanced 2-equation (RNG k-ε) or large eddy simulation (LES) models were implemented since these models gave good results compared to the experimental results. Thus, the two equation (RNG k-ε) mathematical technique was adopted for all numerical cases.

The three-dimensional numerical model was built using the AutoCAD program features and export with (stl.) format then directly import in the Flow-3D software with mesh size of 0.001 m due its accuracy as shown in Figure 6. The boundary conditions were set using a specified pressure at the channel upstream, while it was outflow condition at the channel downstream. The typical view of geometry, boundary conditions and mesh model are shown in Figure 5 and Figure 6, respectively.

---

**Figure 4** RED-Discharge relationship comparison for all models

---

**Figure 5** Boundary condition of the compound models in Flow-3D program
Experimental and Numerical Study of the Effect of the Downstream Spillway Face’s Angle on the Stilling Basin’s Energy Dissipation

Figure 6 Mesh of Finite element model of the compound model

4.2. Numerical Results

The simulation of all physical models was performed and using the same discharge for all models, the hydraulic behavior in the Flow-3D program as shown in Figure 7. Since the third dimension of the model has found to have a minimal effect, the results was focused on the side of the model.

The required depths of water in spillway and stilling basin were taken at the location 1 (depth of water in upstream (y\textsubscript{1})), location 2 (depth of water above spillway (y\textsubscript{c})), location 3 (depth of water at the toe of stilling basin (y\textsubscript{2})), and location 4 (depth of water in downstream) as shown in Figure 8.

![Mesh of Finite element model of the compound model](http://www.iaeme.com/IJCIET/index.asp)
After completing the simulations process and displaying the results, a comparison was made between the experimental and numerical results and it found that there was a very substantial convergence between them with (R2) not less than (0.99) and this is guide of a significant affinity in both laboratory and CFD results as shown in Figure 9.
Experimental and Numerical Study of the Effect of the Downstream Spillway Face’s Angle on the Stilling Basin’s Energy Dissipation

Model (2)

Model (3)

Model (4)

Model (5)

http://www.iaeme.com/IJCIET/index.asp 1335 editor@iaeme.com
5. CONCLUSIONS

This study has presented a new experimental study to investigate the effect of downstream spillway face’s angle on the energy dissipation (ED) over the stilling basin. Six compound physical plywood models for the spillway and stilling basin were built. Additionally, six downstream slope angles were used to study their effect on the ED over stilling basin. A total of 72 experiments runs were conducted and the hydraulic parameters were measured for each run and then the EDs were calculated. The experimental results showed that the relative energy dissipation (RED) of flow on stilling basin is decreased when the discharge is increased and increased when the angle of the downstream face of the spillway is decreased. The numerical results using the FLOW-3D software showed promising results when compared with the experimental results which can consider an effective and economical tool for the design of this type of hydraulic structures.

REFERENCES


Experimental and Numerical Study of the Effect of the Downstream Spillway Face’s Angle on the Stilling Basin’s Energy Dissipation


